

Identify[®] Microsemi Edition Tutorial

March 2015

<http://solvnet.synopsys.com>

SYNOPTSYS[®]

Copyright Notice and Proprietary Information

© 2015 Synopsys, Inc. All Rights Reserved. This software and documentation contain confidential and proprietary information that is the property of Synopsys, Inc. The software and documentation are furnished under a license agreement and may be used or copied only in accordance with the terms of the license agreement. No part of the software and documentation may be reproduced, transmitted, or translated, in any form or by any means, electronic, mechanical, manual, optical, or otherwise, without prior written permission of Synopsys, Inc., or as expressly provided by the license agreement.

Right to Copy Documentation

The license agreement with Synopsys permits licensee to make copies of the documentation for its internal use only.

Each copy shall include all copyrights, trademarks, service marks, and proprietary rights notices, if any. Licensee must assign sequential numbers to all copies. These copies shall contain the following legend on the cover page:

“This document is duplicated with the permission of Synopsys, Inc., for the exclusive use of _____ and its employees. This is copy number _____.”

Destination Control Statement

All technical data contained in this publication is subject to the export control laws of the United States of America. Disclosure to nationals of other countries contrary to United States law is prohibited. It is the reader's responsibility to determine the applicable regulations and to comply with them.

Disclaimer

SYNOPSYS, INC., AND ITS LICENSORS MAKE NO WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, WITH REGARD TO THIS MATERIAL, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

Registered Trademarks (®)

Synopsys, AEON, AMPS, Astro, Behavior Extracting Synthesis Technology, Cadabra, CATS, Certify, CHIPit, CoMET, CODE V, Design Compiler, DesignWare, EMBED-IT!, Formality, Galaxy Custom Designer, Global Synthesis, HAPS, HapsTrak, HDL Analyst, HSIM, HSPICE, Identify, Leda, LightTools, MAST, METeor, ModelTools, NanoSim, NOVeA, OpenVera, ORA, PathMill, Physical Compiler, PrimeTime, SCOPE, Simply Better Results, SiVL, SNUG, SolvNet, Sonic Focus, STAR Memory System, Syndicated, Synplicity, the Synplicity logo, Synplify, Synplify Pro, Synthesis Constraints Optimization Environment, TetraMAX, UMRBus, VCS, Vera, and YIELDirector are registered trademarks of Synopsys, Inc.

Trademarks (™)

AFGen, Apollo, ARC, ASAP, Astro-Rail, Astro-Xtalk, Aurora, AvanWaves, BEST, Columbia, Columbia-CE, Cosmos, CosmosLE, CosmosScope, CRITIC, CustomExplorer, CustomSim, DC Expert, DC Professional, DC Ultra, Design Analyzer, Design Vision, DesignerHDL, DesignPower, DFTMAX, Direct Silicon Access, Discovery, Eclipse, Encore, EPIC, Galaxy, HANEX, HDL Compiler, Hercules, Hierarchical Optimization Technology, High-performance ASIC Prototyping System, HSIMplus, i-Virtual Stepper, IICE, in-Sync, iN-Tandem, Intelli, Jupiter, Jupiter-DP, JupiterXT, JupiterXT-ASIC, Liberty, Libra-Passport, Library Compiler, Macro-PLUS, Magellan, Mars, Mars-Rail, Mars-Xtalk, Milkyway, ModelSource, Module Compiler, MultiPoint, ORAengineering, Physical Analyst, Planet, Planet-PL, Polaris, Power Compiler, Raphael, RippledMixer, Saturn, Scirocco, Scirocco-i, SiWare, Star-RCXT, Star-SimXT, StarRC, System Compiler, System Designer, Taurus, Total-Recall, TSUPREM-4, VCSi, VHDL Compiler, VMC, and Worksheet Buffer are trademarks of Synopsys, Inc.

Service Marks (sm)

MAP-in, SVP Café, and TAP-in are service marks of Synopsys, Inc.

SystemC is a trademark of the Open SystemC Initiative and is used under license.

ARM and AMBA are registered trademarks of ARM Limited.

Saber is a registered trademark of SabreMark Limited Partnership and is used under license.

All other product or company names may be trademarks of their respective owners.

Printed in the U.S.A
March 2015

Contents

Chapter 1: Getting Started

The Debugging System	8
Identify Instrumentor	9
IICE Block	9
Identify Debugger	9
Design Flow	10
Tutorial Requirements	11
Hardware/Software Environments	11

Chapter 2: The Tutorial Design

Design Schematic	14
VHDL Design Files	15
Board Clock Configuration	15

Chapter 3: Instrumenting Your Design

Launching the Identify Instrumentor	17
Creating a Synplify Project	18
Launching the Instrumentor	20
Selecting the Instrumentation	24
Instrumenting the Arbiter	24
Assigning Watchpoints to a Partial Bus	27
Assigning Breakpoints	29
Configuring the IICE	30
Setting the Common IICE Parameters	30
Setting the Individual IICE Parameters	31
Creating a Second IICE	33
Writing the Instrumented Design	35

Chapter 4: Implementing the Design

Synthesis	37
Synthesizing the Bus Demo Design	38
Place and Route	40
Program the Device	40

Chapter 5: Debugging Your Design

Launching the Identify Debugger	42
Specifying the JTAG Cable	43
Setting Up Triggers and Capturing Data	44
Triggering on a Watchpoint	44
Triggering on a Breakpoint	47
Using the Complex Counter	50
Cross Triggering	52
Generating Waveforms	55

CHAPTER 1

Getting Started

The Identify[®] Microsemi[®] Edition tool set includes a software instrumentor and a hardware debugger. The combination of these two tools lets you debug your HDL design:

- In the target system,
- At the target speed,
- At the VHDL/Verilog RTL Source level.

The Identify tools enable the debugging of FPGA designs, FPGA-based prototypes, and system-on-a-chip designs. For the first time, you can debug live hardware using intuitive, HDL-based debugging techniques that provide visibility into the internal operation of your system.

The Identify software easily integrates into your existing design flow so that minimal effort is required to begin the debugging of your HDL designs. To better understand how the Identify instrumentor and debugger work and how they function within your HDL design flow, this guide provides comprehensive information about navigating through the Identify tools and integrating with your other design flow tools.

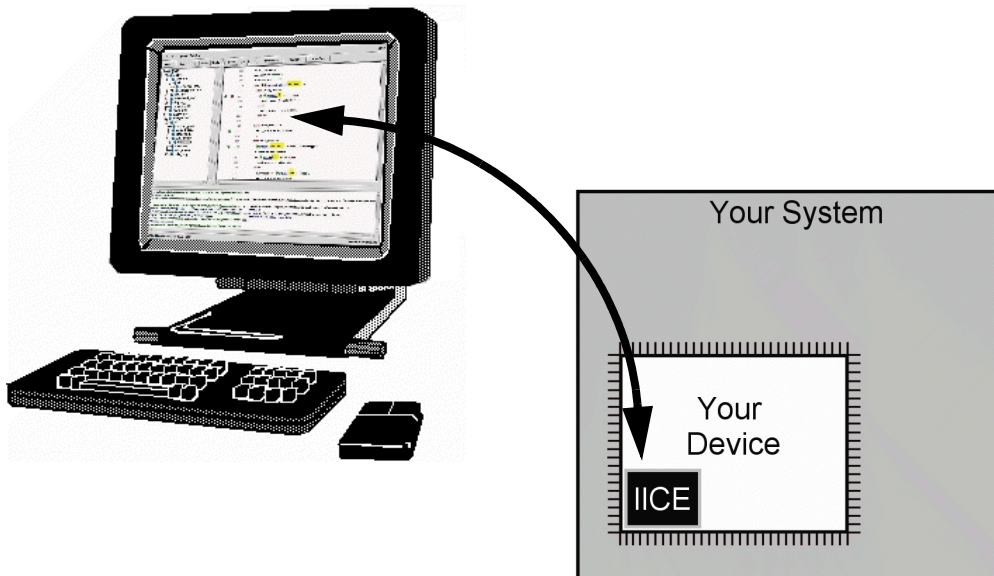
This remainder of this chapter describes:

- [The Debugging System](#)
- [Design Flow](#)
- [Tutorial Requirements](#)

The Debugging System

The Identify concept is based on the principle of in-system debugging. Using the Identify tool set allows you to debug your device in the target system, at target speed while still debugging at the HDL level.

The Identify instrumentor captures your device's internal states by inserting probe hardware (called an IICE™ – Intelligent In-Circuit-Emulator) into your design. The IICE captures internal design states based on user-specified trigger conditions. Data captured at the target device is transferred back to the host computer where it is transformed and displayed by the Identify debugger.



The Identify tool set is a dual-component software system consisting of:

- an Identify instrumentor which inserts and configures the IICE
- an Identify debugger which controls the IICE and displays data from the IICE

The following sections briefly describe these components.

Identify Instrumentor

The Identify instrumentor reads and analyzes the HDL description of a design and provides you with detailed information about the signals that can be sampled and the locations in the source code where breakpoints can be set. This information is collectively referred to as the *instrumentation*.

IICE Block

The Identify instrumentor uses the HDL design files and your selected instrumentation information to create the custom IICE. An IICE block is logic that probes the design and communicates with the Identify user interface through the device JTAG port. The IICE is implemented using the logic resources of the device. The Identify instrumentor then connects one or more IICE blocks to the appropriate locations in the design.

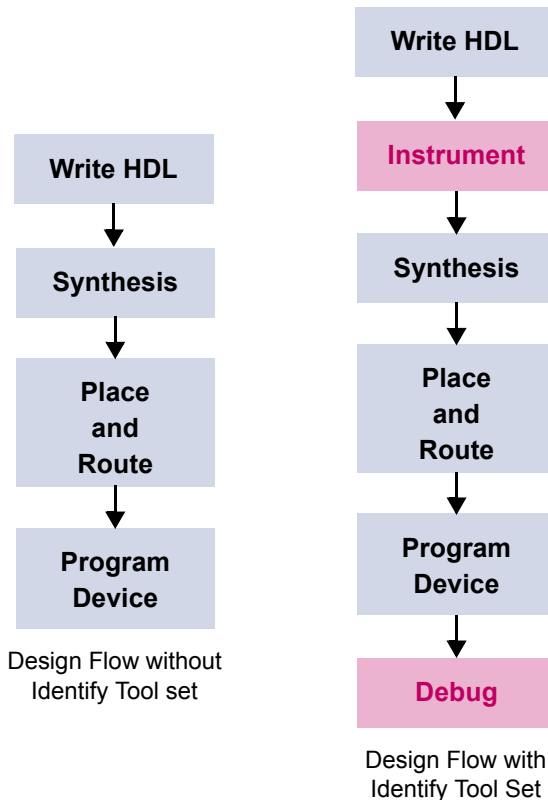
Finally, the Identify instrumentor re-writes your HDL design with the modifications necessary to implement and connect the IICE blocks.

Identify Debugger

The Identify debugger lets you interact with your real hardware at the HDL level. In the Identify debugger, you set trigger conditions to determine when to capture data, and then view the captured data as either annotated source code or as waveforms.

Design Flow

Design flows for HDL design and debugging vary according to the type of hardware and device you use. Displayed below is the typical HDL design flow without the Identify tool set on the left and a typical HDL design flow with the Identify tool set on the right.



In the design flow without the Identify tool set, the first step is to create the HDL source files for the design. Next, the design is synthesized to the target device. Once synthesized, the design is placed and routed before it is finally implemented in the target device.

The design flow with the Identify tool set adds two steps to the standard flow – one step at the beginning and one step at the end. After the HDL source is created, the Identify instrumentor is used to create a debuggable design. This design is then run through the rest of the standard design flow. After the instrumented design has been implemented in the device, the Identify debugger is then used to debug the design in the target system.

Tutorial Requirements

This tutorial guides you through the process of debugging a small HDL design in a real hardware environment. The tutorial teaches you how to generate an instrumented design for debugging and then how to debug that instrumented design.

The tutorial explains how the Identify tools are used in concert with your synthesis and place-and-route flow. However, the tutorial does not provide details and procedures for synthesizing and using place-and-route tools with your instrumented hardware design. For the purposes of this tutorial, it is assumed that you have a working implementation flow. To find out more about these processes, consult your synthesis and place-and-route tool documentation for more information.

Hardware/Software Environments

This tutorial is performed with Microsemi hardware. The tutorial was developed using the following software:

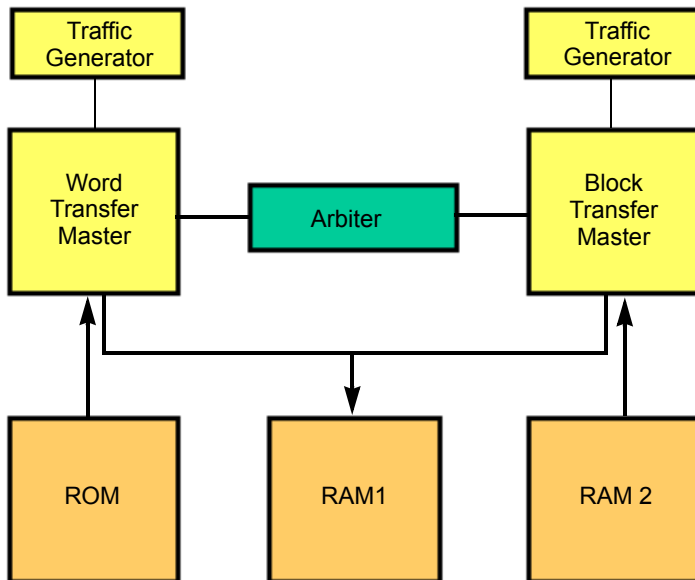
- Identify tool set: Identify J-2014.09M-1 or later
- FPGA synthesis software: Synplify Pro® J-2014-09M-SP2 or later
- Place-and-Route software: Libero v10.1

Note: Other device families and devices, and board systems can be used.

CHAPTER 2

The Tutorial Design

The Identify tool set can debug a variety of HDL designs. To better understand the debugging process, this guide provides a small HDL design example. The example design is a simple bus demo circuit. The HDL source is written in VHDL and is based on the open-source Wishbone bus. A block diagram of the design is shown below.



The design contains two bus masters connected to three slave devices. The bus masters are the Word Transfer Master and the Block Transfer Master. The slaves are the three memory devices, ROM, RAM1, and RAM2. The bus masters are driven by traffic generators that request random bus transactions. The arbiter controls master access to prevent bus conflicts. The arbiter monitors bus requests, and grants bus accesses based on a fair algorithm.

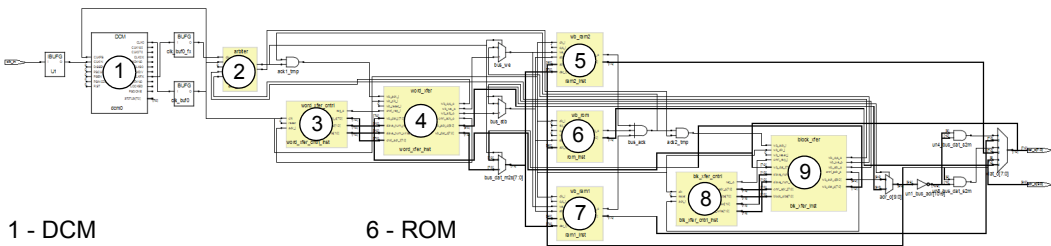
The arbiter uses two clocks. One clock is used by the state machine and the other is used by the other logic modules and also to synchronize with the state-machine clock. The two clocks are used to demonstrate cross-triggering as explained later in this tutorial. The focus of the tutorial is on the arbiter and a 4-bit counter.

For the tutorial, you will instrument the design for an IGLOO PLUS device. You will then debug the design hardware to gain a basic understanding of the Identify tool set and how it integrates into your synthesis environment.

Note: Other supported Microsemi device families and devices can be used.

Design Schematic

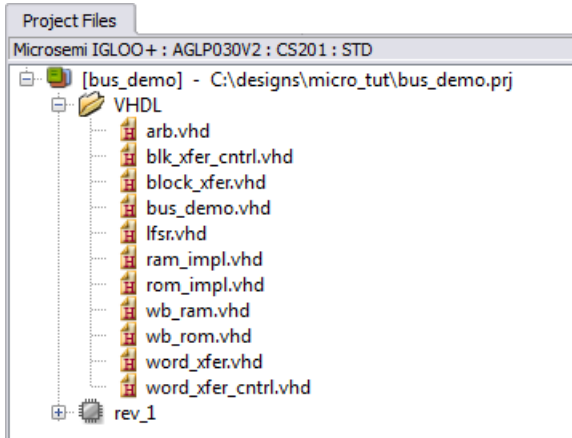
The following figure is a schematic representation of the simple bus circuit used for the tutorial design.



- | | |
|------------------------------|-------------------------------|
| 1 - DCM | 6 - ROM |
| 2 - Arbiter | 7 - RAM 1 |
| 3 - Word Transfer Controller | 8 - Block Transfer Controller |
| 4 - Word Transfer Logic | 9 - Block Transfer Logic |
| 5 - RAM 2 | |

VHDL Design Files

The bus demo design used for this tutorial is implemented in VHDL. The initial design files, as loaded into Synplify Pro, are shown in the figure below:



Board Clock Configuration

As previously mentioned, this tutorial was developed using a Microsemi IGLOO PLUS device (AGLP030V2). The bus_demo.vhd file in the tutorial design uses a 50 MHz clock (clk) and a 35 MHz clock (clk_arb) for the design. These frequencies are set by create_clock constraints in the constraint file provided with the tutorial.

CHAPTER 3

Instrumenting Your Design

The Identify instrumentor selects the design visibility (breakpoints and watchpoints) and special hardware configurations including complex event counters, and sampling and triggering modes for your design. The goal of the instrumentation process is to define an IICE and insert it into your HDL design. The instrumentation flow is:

- [Launching the Identify Instrumentor](#)
- [Selecting the Instrumentation](#)
- [Configuring the IICE](#)
- [Writing the Instrumented Design](#)

The HDL design files for the tutorial are included in a tar or zip file in the share subdirectory within the installation directory. Unpack the appropriate file to a local directory and make sure that you have read and write permission for both the directory and files.

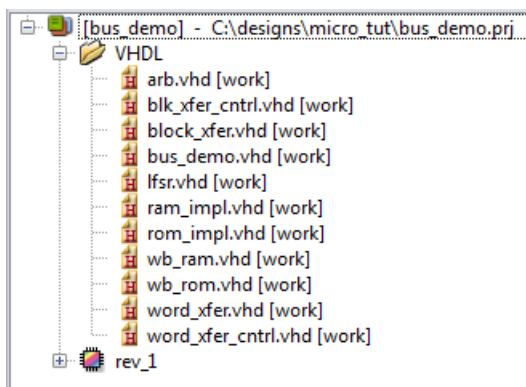
Launching the Identify Instrumentor

The Identify instrumentor is launched directly from the Synplify Pro synthesis tool.

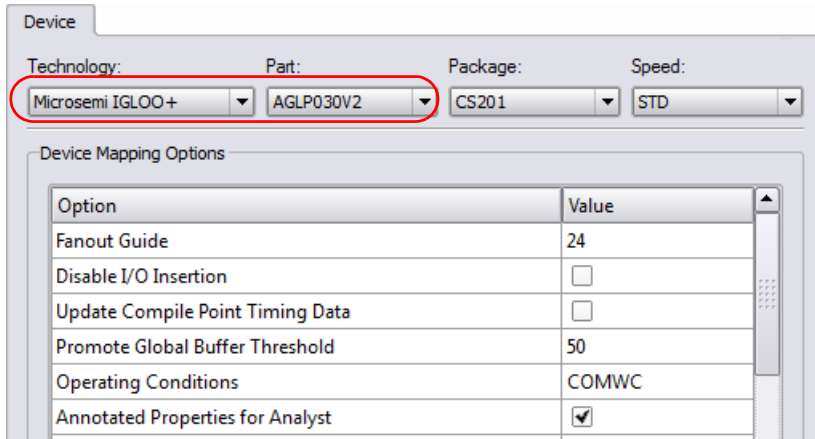
Creating a Synplify Project

Before you can launch the Identify instrumentor, you must create the project in Synplify Pro. To create a project:

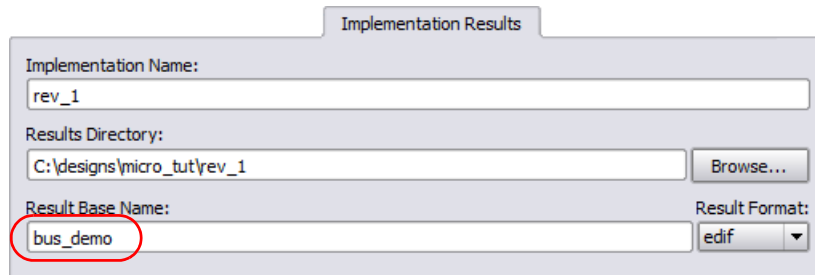
1. Start the Synplify Pro synthesis tool.
2. Select the Open Project button (just below the large Run button) and click New Project in the dialog box.
3. Click File->Save As from the top menu bar and navigate to the directory where you downloaded the design files. Enter bus_demo.prj as the project name and click Save.
4. Click the Add File button to display the Add Files to Project dialog box. In the dialog box, click <-Add All to select all of the HDL files and then click OK to add the selected files to the project and close the dialog box. If the list of files is not displayed in the dialog box, make sure that Files of type field is set to HDL Files (or All Files).
5. Expand the VHDL (shown) or Verilog folder by clicking on the “+” to the right of the folder to display the list of design files as show below.



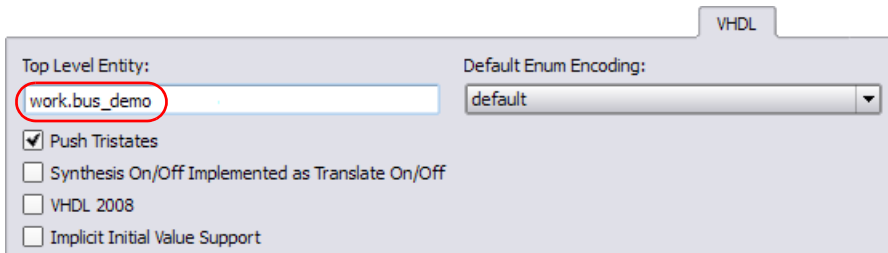
- From the top menu bar, select Project->Implementation Options and, if necessary, select the Device tab. Set the technology in the Technology drop-down menu to Microsemi IGLOO+ and set the device in the adjacent Part drop-down menu to AGLP030V2.



- Without closing the Implementation Options dialog box, select the Implementation Results tab and make sure that the Result Base Name is bus_demo and that the Result Format is edif (the only selection).



- Select the VHDL tab and enter work.bus_demo in the Top Level Entity field. Note that it is recommended for the top-level VHDL entity to be in the work library.

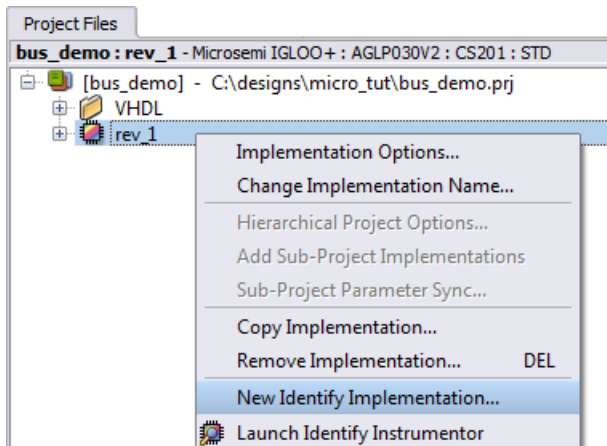


9. Click OK to close the Implementation Options dialog box.

Launching the Instrumentor

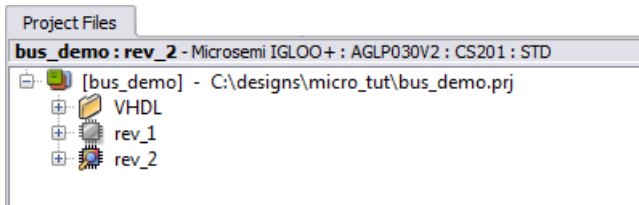
After creating a project in Synplify Pro, you are ready to launch the Identify instrumentor directly from your synthesis project. To launch the Identify instrumentor:

1. Right click on the rev_1 implementation in the project view and select New Identify Implementation from the popup menu.

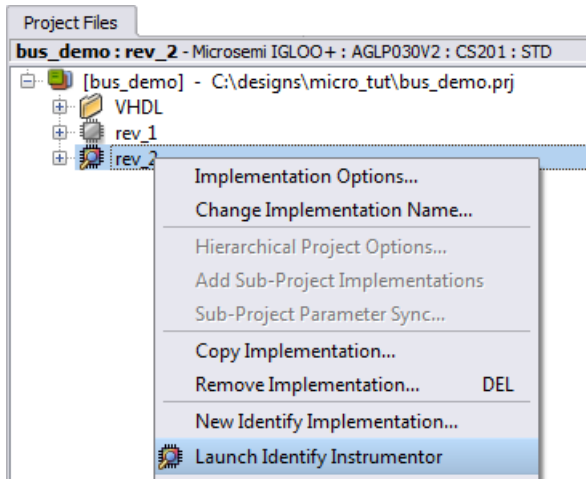


2. When you select New Identify Implementation, you are presented with the Add Identify Implementation dialog box. This dialog box is identical to the Implementation Options dialog box that you used when setting up your project. Unless you have any additional changes (unnecessary for the tutorial), click OK to close the dialog box.

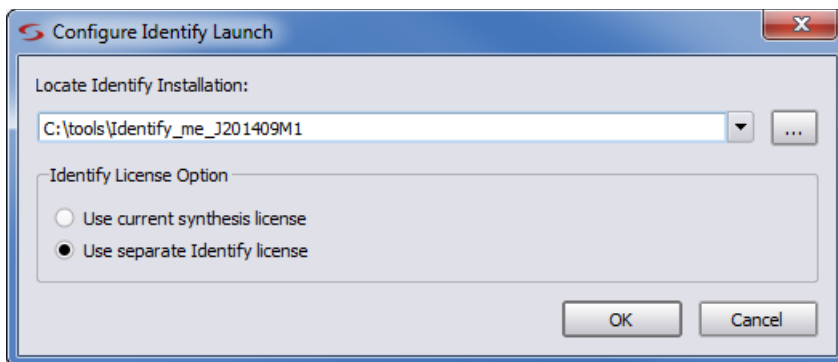
- When you close the dialog box, an Identify implementation is added to the project as noted by the icon with the magnifying glass.



- Right click on the Identify implementation and select Launch Identify Instrumentor from the popup menu.

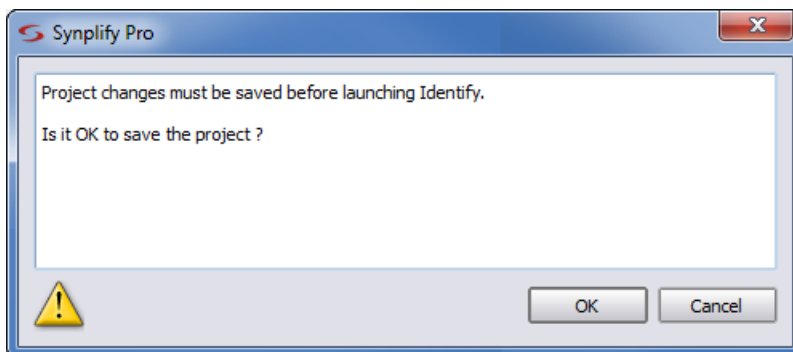


If the synthesis tool detects more than one copy of Identify or if the path to the Identify executable has not been established, the following dialog box appears.



Click to OK to accept the executable currently displayed or click the browser and navigate to the Identify installation directory.

5. You are next prompted to save the changes to your synthesis tool.



Click OK to save the project and to launch the Identify instrumentor.

Launching the Identify instrumentor from the Synplify Pro synthesis tool:

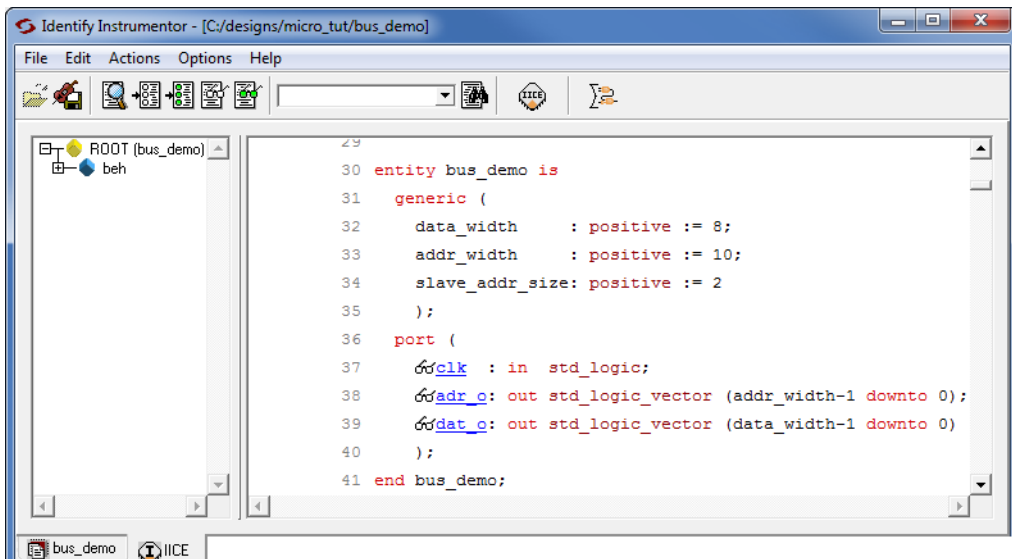
- Brings up the Identify instrumentor graphical interface.

Note: If you are prompted to select a license type, select the appropriate type from the list; check the Save as default license type box to avoid the license prompt in future sessions.

- Opens the synthesis project (`prj`) file in the Identify instrumentor. The list of design files, their work directories, and the device family are automatically extracted from this `prj` file.
- Automatically compiles the design files in the specified technology (IGLOO+) using the Synplify Pro compiler.
- Displays the design hierarchy and the HDL file content with the potential watchpoints and breakpoints identified for selection.

Selecting the Instrumentation

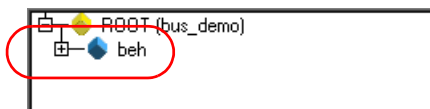
After your VHDL design compiles, the instrumentation window displays the top-level entity VHDL code on the right and the hierarchy browser on the left (if the instrumentation window is not displayed, click on the “IICE” tab at the bottom of the project window). The tutorial uses the hierarchy browser to navigate through your design. Clicking on a hierarchical node updates the corresponding section of code in the source code display on the right.



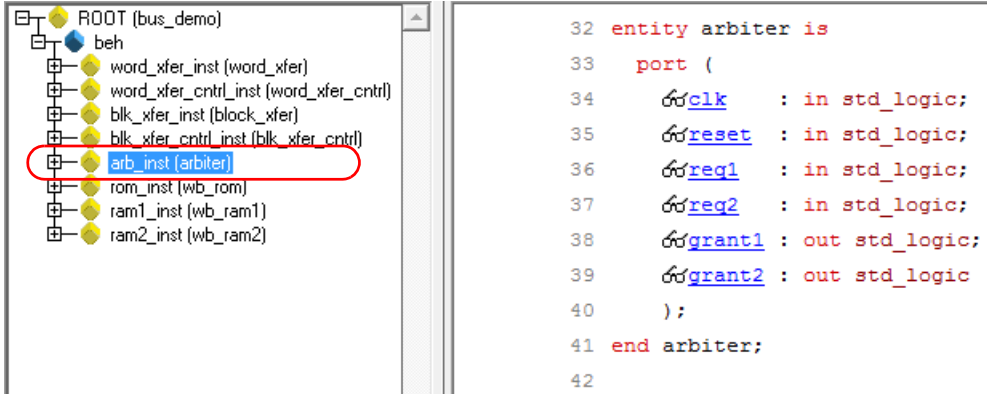
Instrumenting the Arbiter

To instrument the arbiter:

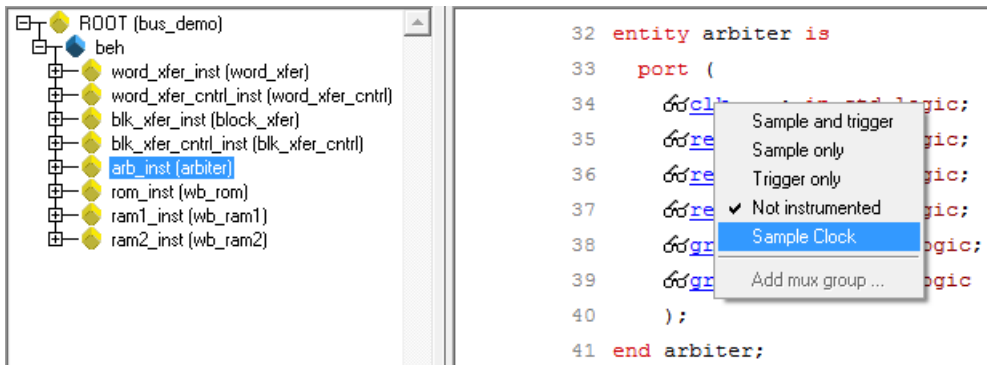
1. Expand the beh block in the hierarchy browser by clicking on the ‘+’ sign adjacent to the block.



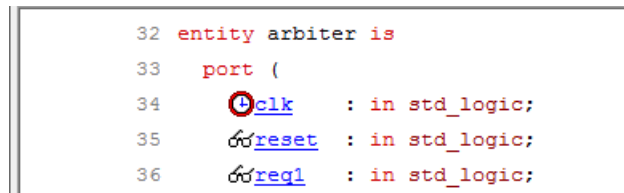
2. In the expanded hierarchy, click on arb_inst (arbiter) to update the instrumentation window with the arbiter entity beginning on line 32.



3. With the arbiter still displayed, click on the clk signal on line 34 (or the adjacent eyeglass icon) to display the watchpoint menu.



4. In the watchpoint menu, click Sample Clock. The eyeglass icon is replaced with the sample clock icon to indicate that the clk signal is now defined as the sample clock for the IICE.



- Locate the first occurrence of the `curr_state` signal on line 45 in the source code display. Select this signal for instrumentation by clicking on the signal name (or the adjacent eyeglass icon) and then selecting **Sample and Trigger** from the popup menu. The eyeglass icons preceding each occurrence of the `curr_state` signal in the VHDL code for the arbiter will be green.

When you select a watchpoint instrumentation type from the popup menu, the icon lenses change color according to the type selected as outlined in the following table.

Icon Color	Watchpoint Selection
Green	Sample and trigger
Blue	Sample only
Red	Trigger only
Clear (unfilled)	Not instrumented

The command that you would enter manually or in a shell script to define the watchpoint signal (signals add `-sample -trigger -iice IICE /beh/arb_inst/beh/curr_state`) and the cumulative instrumentation to implement the watchpoint are displayed in the console window.

- To become familiar with the command-line interface, use the console window to set a sample and trigger watchpoint on the `next_state` signal. At the command prompt, enter:

```
signals add -sample -trigger -iice IICE
           /beh/arb_inst/beh/next_state
```

Check that the eyeglass icon preceding the `next_state` signal on line 45 is green (sample and trigger).

- Again using the graphical interface, locate signal `grant1` on line 38 and set its watchpoint to **Sample Only**. The icon lenses are blue for **Sample Only**.
- Locate signal `grant2` on line 39 and set its watchpoint to **Trigger Only**. The icon lenses are red for **Trigger Only**.

The following figure shows the resulting assigned watchpoints.

```

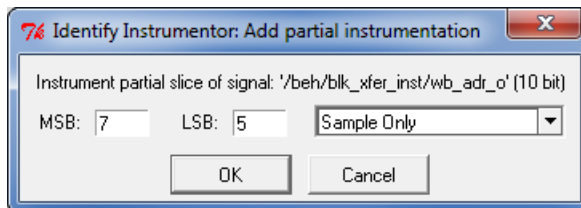
32 entity arbiter is
33   port (
34     clk      : in std_logic;
35     reset    : in std_logic;
36     req1     : in std_logic;
37     req2     : in std_logic;
38     grant1   : out std_logic;
39     grant2   : out std_logic
40   );
41 end arbiter;
42
43 architecture beh of arbiter is
44   type states is ( st_idle1, st_idle2, st_grant1, st_grant2);
45   signal curr_state, next_state: states;

```

Assigning Watchpoints to a Partial Bus

This section shows you how to instrument a partial bus. The section uses the blk_xfer_inst (block_xfer) entity from the tutorial design to illustrate watchpoint assignment. To instrument the partial bus:

1. In the hierarchy browser, click on blk_xfer_inst (block_xfer) to update the instrumentation window with the block_xfer entity.
2. Click on the wb_adr_o bus on line 35 and use the popup menu to select Add partial instrumentation.
3. In the dialog box displayed, enter 7 for the MSB value, enter 5 for the LSB value, and select Sample Only from the drop-down menu for the instrumentation type.



- Click OK. The eyeglass icon is replaced with a large P symbol to indicate a partially assigned bus. The symbol is blue to indicate sample only; other sample options are designated by other colors.

```

28 entity block_xfer is
29   generic (
30     data_width: positive := 8;
31     slave_addr_size: positive := 2;
32     addr_width: positive := 10
33   );
34   port (
35     P wb_adr_o: out std_logic_vector (addr_width-1 downto 0) :=
36     wb_dat_i: in std_logic_vector (data_width-1 downto 0) := (
37     wb_dat_o: out std_logic_vector (data_width-1 downto 0) := (
38     wb_cyc_o: out std_logic;

```

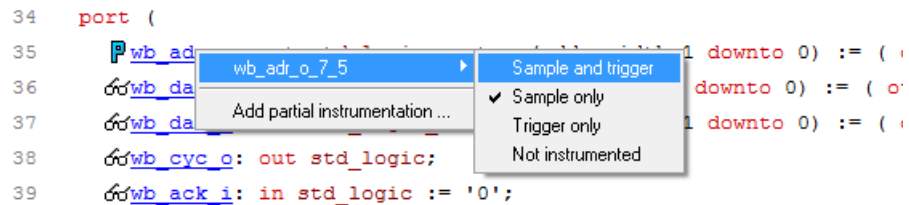
Note: A single bit of a bus can be instrumented by designating it as both the LSB and the MSB.

- Again click on the wb_adr_o bus and note that the previous bit selection is displayed and that you can change the instrumentation options for that selection.

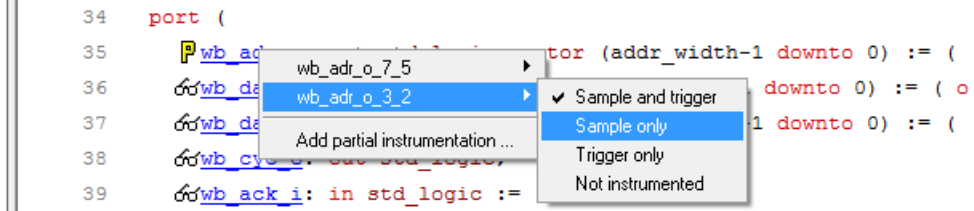
```

34   port (
35     P wb_adr_o: out std_logic_vector (addr_width-1 downto 0) := (
36     wb_dat_i: in std_logic_vector (data_width-1 downto 0) := (
37     wb_dat_o: out std_logic_vector (data_width-1 downto 0) := (
38     wb_cyc_o: out std_logic;
39     wb_ack_i: in std_logic := '0';

```



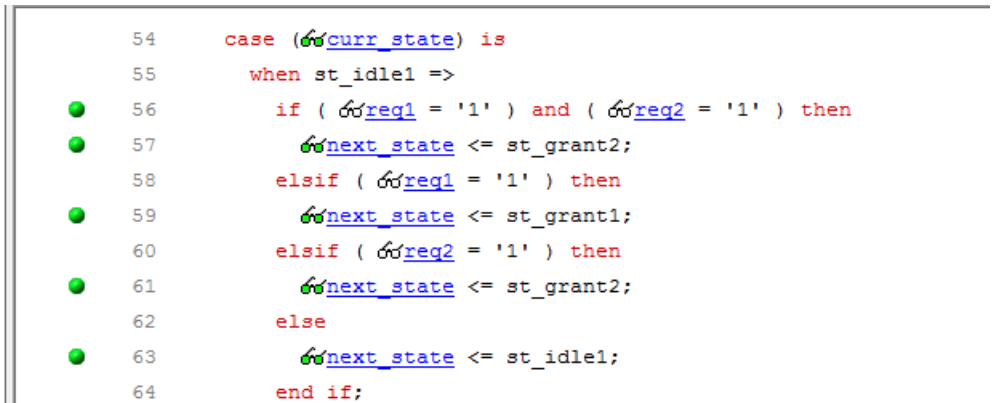
- Select Add partial instrumentation and, in the dialog box, set MSB to 3, set LSB to 2, and leave the instrumentation options set to Sample and trigger. Click OK and note that the color of the P icon changes to yellow to indicate mixed instrumentation types (Sample only and Sample and trigger).
- Click on the wb_adr_o bus to see both partial instrumentations. Either can be changed from one type to another (or “not instrumented”) using the popup menu.



Assigning Breakpoints

Breakpoints are used to trigger on reaching a branch in the code rather than triggering on a logic-level transition. For this portion of the tutorial, the arbiter entity is again used to illustrate breakpoint assignment. To assign breakpoints:

1. Again click on arb_inst (arbiter) in the hierarchy browser to update the instrumentation window with the arbiter entity.
2. In the source code display, scroll down to the case statement beginning on line 54. Click on the first five breakpoint buttons in the left margin on lines 56, 57, 59, 61, and 63. The buttons change from clear to green to indicate that they are active.



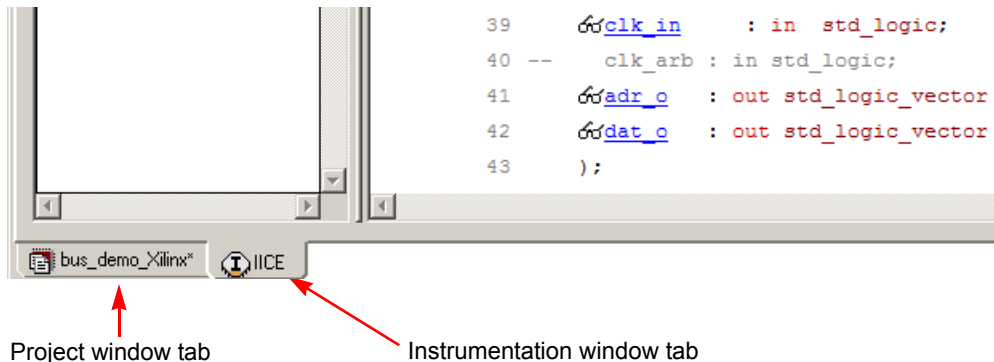
Similar to the watchpoint selection, the Identify instrumentor displays the commands that you would enter manually or in a shell script to define the breakpoints.

Configuring the IICE

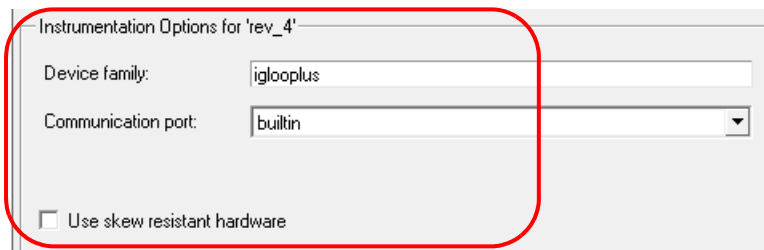
The IICE settings common to all IICE units for an instrumentation are set in the project window, and the individual IICE settings unique to each IICE in a multi-IICE configuration are set on the IICE Configuration dialog box.

Setting the Common IICE Parameters

The common IICE parameters are set in the project window. To redisplay the project window, click the project window tab (bus_demo) in the lower left corner of the instrumentation window.



The common IICE parameters appear in the Implementation Options block on the right side of the project window.



For the tutorial design:

- The device family (iglooplus) is specified in the synthesis tool and reported in the read-only Device family field from the -technology entry in the project file.

- Communication between the Identify debugger and the hardware is specified by the Communication port field. For the tutorial, make sure that builtin is specified.
- Leave the Use skew resistant hardware check box unchecked.

Setting the Individual IICE Parameters



The individual IICE parameters are set by tabs on the IICE Configuration dialog box. Click on the Edit IICE settings icon on the toolbar or select Actions->Configure IICE from the menu to bring up the IICE Sampler tab shown in the following figure.

The screenshot shows the 'IICE Sampler' configuration dialog box. At the top, there is a tab labeled 'IICE Sampler'. Below the tab, there are two dropdown menus: 'Current IICE:' set to 'IICE' and 'IICE type:' set to 'regular'. The main area is divided into three sections. The first section, 'IICE Sampler', contains a 'Buffer type:' dropdown set to 'internal_memory', a 'Sample depth:' spinner set to '2048' (circled in red), and three unchecked checkboxes: 'Allow qualified sampling', 'Allow always-armed triggering', and 'Allow data compression'. The second section, 'Sample Clock', contains a 'Sample clock:' text field with the value '/beh/arb_inst/clk' and a 'Clock edge:' section with two radio buttons: 'Positive' (selected) and 'Negative'.

The IICE Sampler tab defines the current IICE unit, buffer type, sample depth of the data sampling hardware, controls the three optional sampling modes, and defines the sample clock and clock edge.

For the tutorial design:

- Leave the Buffer type set to internal_memory (only available selection).
- Set the Sample depth to 2048 (use the up/down arrows).
- Leave the Allow qualified sampling check box unchecked.

- Leave the Allow always-armed sampling check box unchecked.
- Leave the Allow data compression check box unchecked.
- Make sure that the Sample clock specifies /beh/arb_inst/clk and that Positive is selected for Clock edge. Note that the sample clock was instrumented at the beginning of the tutorial; to change the clock edge, you must use the IICE Sampler tab.

After you have set and/or verified the above IICE Sampler tab settings, click the IICE Controller tab.

The screenshot shows the IICE Controller configuration window. At the top, there is a tab labeled "IICE Controller". Below the tab, there are two dropdown menus: "Current IICE:" set to "IICE" and "IICE type:" set to "regular". The main configuration area is divided into two sections: "IICE Controller" and "IICE Options".

IICE Controller

- Simple triggering
- Complex counter triggering
 - Width: 16
- State Machine triggering
 - Trigger states: 4
 - Trigger conditions: 4
 - Counter width: 16

IICE Options

- Import external trigger signals: 0
- Export IICE trigger signal
- Allow cross-triggering in IICE

On the IICE Controller tab:

- Make sure that the Complex counter triggering radio button is selected and that the Width is set to 16.
- Leave the Import external trigger signals field set to 0 (disabled).
- Leave the Export IICE trigger signal check box unchecked.
- The Allow cross-triggering in IICE selection is currently disabled (grayed-out); this selection will be enabled when a second IICE unit is added as described in the next section.

When all of the IICE configuration settings have been made or verified, click the OK button at the bottom of the dialog box.

Creating a Second IICE

A second IICE unit is used with the tutorial to create a trigger event to illustrate cross triggering. Identify supports multiple instances of IICE units to allow multiple instrumentations of a design to exist concurrently. The individual instrumentations do not interfere with one another and support triggering in different clock domains. To create a second IICE unit:

1. Redisplay the project window by clicking the project window tab (`bus_demo`) in the lower left corner of the instrumentation window.
2. Click the New IICE button to add the second IICE unit (`IICE_0`). When you click the button, the instrumentation window is redisplayed with the second IICE tab (`IICE_0`) active and with all of the watchpoints and breakpoints unselected.
3. Expand the `beh` block in the hierarchy browser by clicking on the '+' sign adjacent to the block and click `blk_xfer_inst` (`block_xfer`).
4. Instrument the `wb_clk_i` signal on line 43 as the sample clock.
5. Instrument the `wb_cyc_o` signal on line 38 for sample and trigger.
6. Click the Edit IICE settings icon in the toolbar to open the Configure IICE parameters dialog box.
7. On the IICE Sampler tab, make sure that the Sample depth is set to 128. Note that the Current IICE field reports `IICE_0` and that the sample clock is `/beh/blk_xfer_inst/wb_clk_i`.

IICE Sampler

Current IICE: IICE_0 IICE type: regular

IICE Sampler

Buffer type: internal_memory

Sample depth: 128

Allow qualified sampling

Allow always-armed triggering

Allow data compression

Sample Clock

Sample clock: /beh/blk_xfer_inst/wb_clk_i

Clock edge: Positive Negative

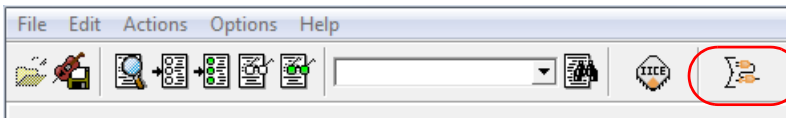
8. Select the IICE Controller tab and make sure that the Simple triggering radio button is selected.
9. Make sure that the Allow cross-triggering in IICE check box is checked and then click OK to close both tabs.

Writing the Instrumented Design



To write the instrumented design, select File->Save project instrumentation from the menu or click on the Save project's activated instrumentation icon on the toolbar. Saving the project automatically adds a set of files to the Identify implementation directory (rev_2). These files are subsequently used by the Synplify Pro synthesis tool to insert the instrumented logic into the design netlist.

The information at the bottom of the display reports the total number of instrumented signals to implement in each IICE. To display the FPGA resource estimates for the selected IICE unit, click the IICEs resource estimation icon.



After several seconds, an estimate of the additional resource requirements is reported at the bottom of the display.

```
C:/designs/micro_tut$ device estimate -resources -iice IICE_0
INFO: device estimate -resources -iice IICE_0
INFO:   IICE=IICE_0
           Resources estimated: 0 BRams, 860 CoreCells
```

The tutorial implements a small design on a large FPGA, and the additional IICE resources are not an issue. Implementing designs that approach the capacity of an FPGA can limit the amount of instrumentation logic (watch-points and breakpoints) that can be included.

For the tutorial, the IICE is configured to use the built-in communications mode. If you had selected the soft JTAG port setting, four extra ports (three input and one output) would have been added to your design. If this was the case, you would have to provide access to these ports on your board to connect the JTAG communication cable to these ports (see the *User Guide* for more information).

This concludes the instrumentation phase of the tutorial. You can now close the Identify instrumentor.

CHAPTER 4

Implementing the Design

At this time, you should have successfully completed the instrumentation of the tutorial design and saved the instrumented project.

Implementing the design involves the following steps:

- Synthesizing the instrumented VHDL or Verilog
- Placing and routing the design to produce a bit file
- Programming the device with the bit file

Synthesis translates the instrumented VHDL or Verilog code to a mapped netlist. Place-and-route tools further process this EDIF netlist to create the implementation of the instrumented design.

Synthesis

The Identify Microsemi Edition tool set is explicitly designed to work with the Synplify Pro synthesis tool. When you save the activated instrumentation for your project at the end of the instrumentation phase (and run synthesis), several new files are created. These files include:

- `identify.idc` – this file is created in the implementation (`rev_2`) directory and describes instrumentation design constraints. In the synthesis tool, the pre-mapper reads the constraints from this file to insert the instrumentation logic into the design.

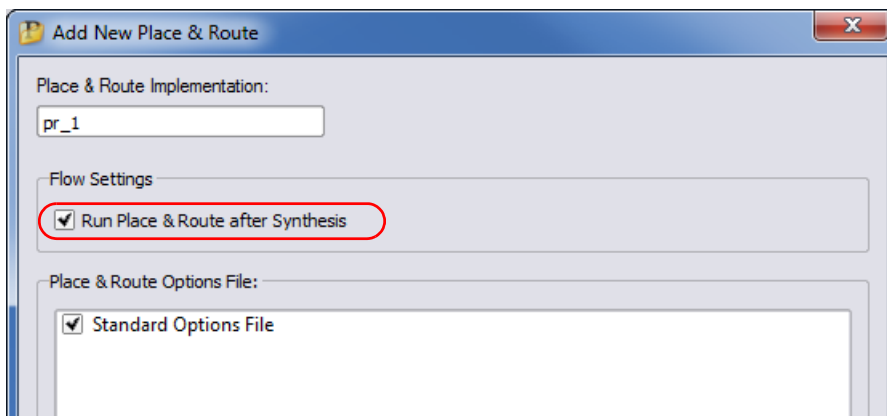
- `syn_dics.cdc` – this file is created in the `instr_sources` subdirectory in the implementation directory and consists of compiler design constraints specific to the instrumentation. At the start of compilation in the synthesis tool, the compiler reads the constraints and adds the appropriate hyper-source attributes to the gate-level netlist file.
- `syn_dics.sdc` – this file is created in the `instr_sources` subdirectory in the implementation directory and includes the timing constraints for the instrumentation logic.

Unlike previous Identify releases, the instrumented logic is not inserted into the RTL netlist and is only visible in the gate-level netlist.

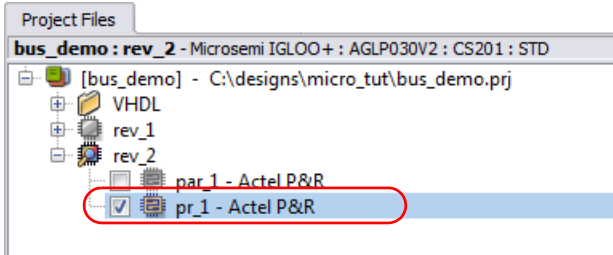
Synthesizing the Bus Demo Design

To synthesize the instrumented bus-demo design:

1. In the Synplify Pro project view, right click on the identify implementation (`rev_2`) and select **Add Place & Route** from the popup menu to display the **Add New Place & Route Task** dialog box.
2. In the dialog box, make sure that **Run Place & Route after synthesis** is checked and click **OK**

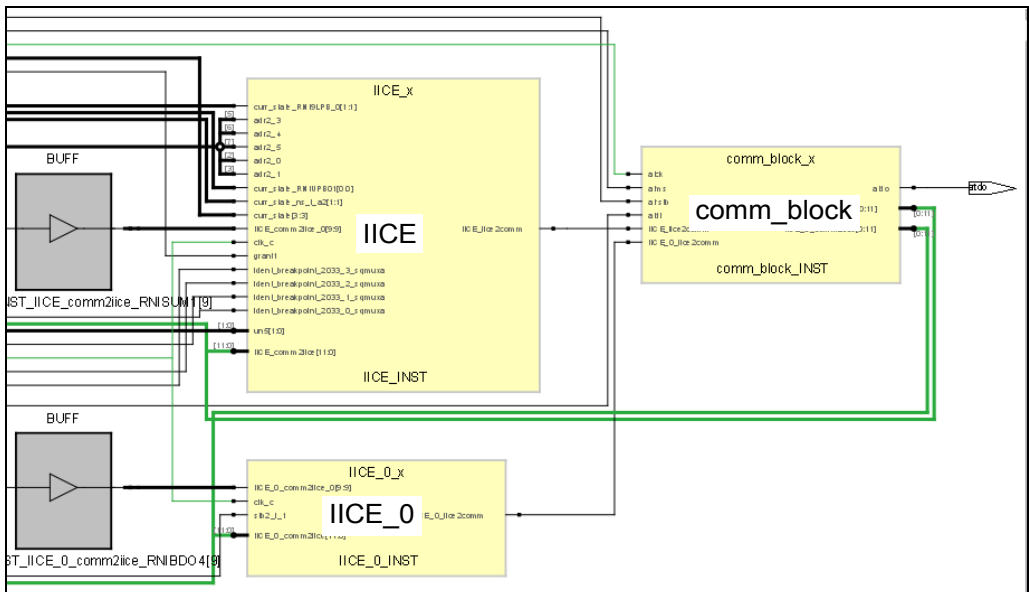


3. In the project view, make sure that `pr_1 - Actel P&R` is checked in the Identify implementation. The following figure shows the final project view with the added file.



4. Click the Run button or select Run->Run from the menu bar to synthesize the tutorial design. When prompted to save the changes to your project, click Yes.

When synthesis is complete, you can view the inserted comm and IICE blocks in the Technology (srm) view.



Place and Route

After synthesis, run the resulting (EDIF) netlist file generated by the Synplify Pro synthesis tool through the Libero place and route tool. Running the file of the instrumented design through place and route is identical to running place and route on the original design. Refer to your place and route tool documentation for further information.

If you selected the soft JTAG port setting, four external ports are added to the tutorial design. These ports are:

- identify_jtag_tdi (input serial data IN signal.)
- identify_jtag_tck (input asynchronous clock signal)
- identify_jtag_tms (input control signal)
- identify_jtag_tdo (output serial data OUT signal.)

You must provide proper pin locations for these ports to your place and route tools. These pins must be connected to your JTAG cable to enable communication with the Identify debugger. Please refer to the user guide for more information about this process.

Program the Device

Programming the target device with the bit file of the instrumented design is identical to programming the target device with the original design's bit file. Please refer to your programming tool documentation for further information.

CHAPTER 5

Debugging Your Design

This chapter shows you how to debug the tutorial design you instrumented earlier. To proceed with debugging, your design must be synthesized, run through place and route, and programmed into the FPGA device. Also, the device must be connected to the host machine using the proper cable and pin connections.

The debugging process consists of setting up the debugging environment, enabling triggers in the IICE, and examining the data downloaded from your hardware. The debug flow is:

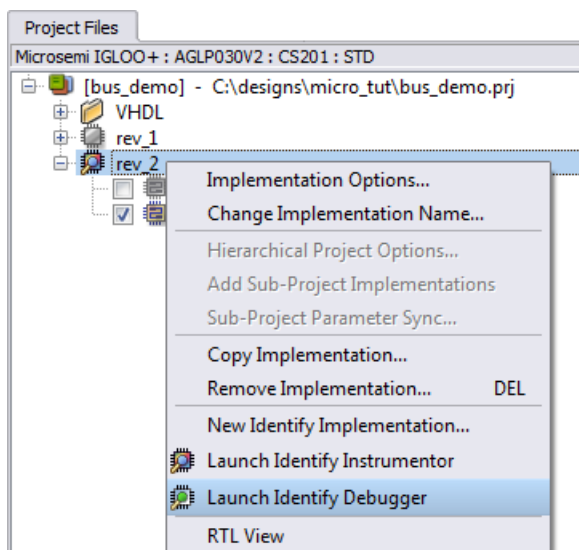
- [Launching the Identify Debugger](#)
- [Specifying the JTAG Cable](#)
- [Setting Up Triggers and Capturing Data](#)
- [Generating Waveforms](#)

Launching the Identify Debugger

You can launch the Identify debugger directly from the Synplify Pro synthesis tool or you can open the Identify debugger from the GUI and then use the Identify debugger Open Project File dialog box to open your synthesis project. Launching the debugger from the Synplify Pro synthesis tool simplifies the procedure by using the currently active Identify implementation.

To launch the Identify debugger from the Synplify Pro synthesis tool:

1. Open the `bus_demo` project in the Synplify Pro synthesis tool and highlight the Identify implementation in the Project view.
2. With the right mouse button, select **Launch Identify Debugger** from the popup menu or click the **Launch Identify Debugger** icon in the top menu bar.



Launching the Identify debugger from the Synplify Pro synthesis tool:

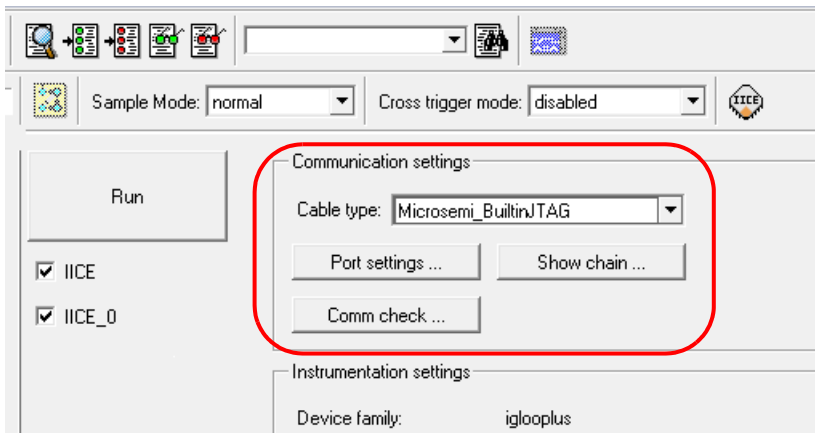
- Brings up the Identify debugger graphical interface.
- Displays the design hierarchy and the HDL file content with the previously instrumented watchpoints and breakpoints visible.

Only the instrumentation that you entered previously with the Identify instrumentor is displayed in the Identify debugger.

Specifying the JTAG Cable

The type of cable used to communicate with the FPGA must be specified so that the correct protocol is used to communicate with the physical device. The JTAG cable and cable port are set in the Communication settings section of the project window after the project is read in. To restore this window, click the bus_demo tab at the bottom of the window.

The hardware used to develop this tutorial used a flashPro USB cable – your cable type may differ. The Microsemi_BuiltinJTAG setting enables you to use any of the flashPro cables.



After completing the computer-to-board cable interface, use the com check command to verify that the JTAG cable is connected properly and that the Identify debugger recognizes the instrumented design in the JTAG chain. For more information, see Chapter 9, *Connecting to the Target System*, in the *User Guide*.

Setting Up Triggers and Capturing Data

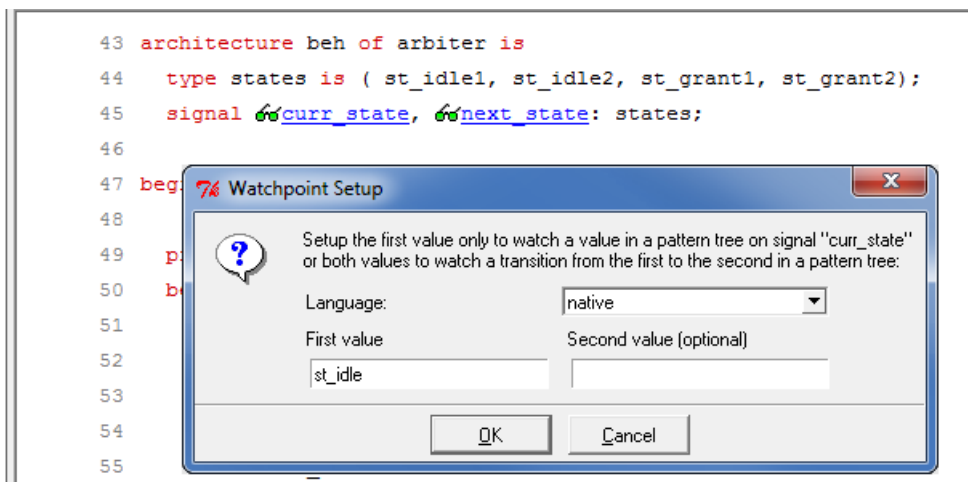
Trigger operations include:

- [Triggering on a Watchpoint](#)
- [Triggering on a Breakpoint](#)
- [Using the Complex Counter](#)
- [Cross Triggering](#)

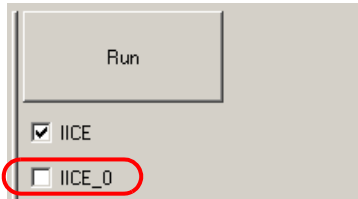
Triggering on a Watchpoint

A watchpoint trigger can be specified on any sampled signal. The Watchpoint Setup dialog box contains a full HDL parser so that any legal VHDL or Verilog expression that evaluates to a constant can be used. When the signal with the watchpoint trigger changes to the value of the watchpoint expression, the Identify debugger triggers. To set a simple watchpoint:

1. Select IICE unit IICE, expand the beh icon, and click on arb_inst (arbiter) to update the instrumentation window with the arbiter entity.
2. Click on curr_state signal on line 45 and select Set trigger expressions from the popup menu.
3. In the first value field, enter st_idle1, and leave the second field blank.



- Click OK to close the Watchpoint Setup dialog box. The color of the curr_state watchpoint icon changes from green to red to indicate that it is active.
- Redisplay the project view (click the bus_demo tab) and uncheck the IICE_0 selection under the Run button.



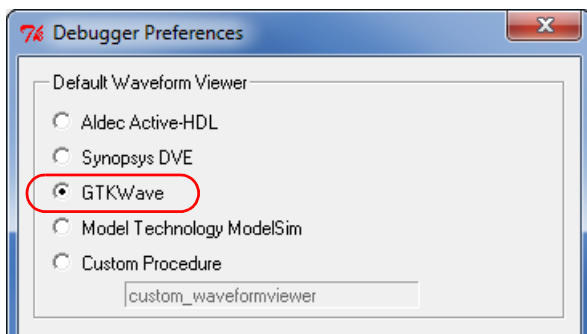
- Click the Run button to arm the IICE on the FPGA device.
- When the trigger occurs, the sampled data is transferred back to the debugger (the icon in the IICE tab changes to a green check mark coincident with the trigger). Click the IICE tab to redisplay the instrumentation window. The sampled data is displayed in the source code display next to the sampled signals. Notice that the values in yellow adjacent to the sampled signals are the data sampled from the FPGA.

```

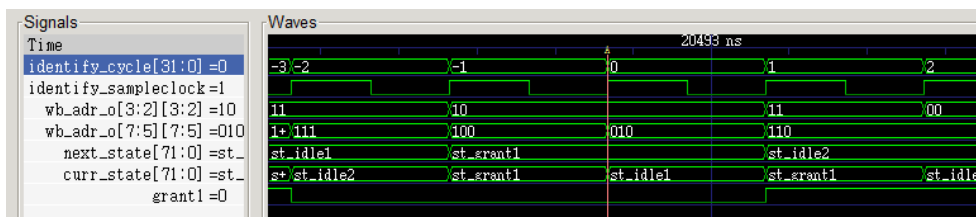
49  process ( curr_statest_idle1, req1, req2 )
50  begin
51      grant1'0' <= '0';
52      grant2 <= '0';
53
54      case (curr_statest_idle1) is
55          when st_idle1 =>
56              if ( req1 = '1' ) and ( req2 = '1' ) then
57                  next_statest_grant1 <= st_grant2;
58              elsif ( req1 = '1' ) then
59                  next_statest_grant1 <= st_grant1;
60              elsif ( req2 = '1' ) then
61                  next_statest_grant1 <= st_grant2;
62              else
63                  next_statest_grant1 <= st_idle1;
64              end if;

```

- This step uses a waveform viewer. The tutorial was developed on a Windows platform that used the GTKWave viewer. Open the Debugger Preferences dialog box (select Options-> Debugger preferences) and make sure that the GTKWave radio button is selected. Be sure to click OK if it was necessary to change the setting.



- Click the Open Waveform Display icon to open the waveform viewer. The waveform shows the extra debugger signals (identify_cycle and identify_sampleclock) and the watchpointed signals. In the waveform diagram, the partial bus waveforms are displayed as separate entries; the display is divided evenly between events before and after the trigger point (the red center line in the diagram).



- Close the GTKWave waveform viewer.

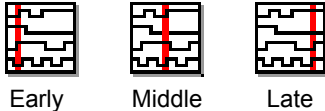
- From the hierarchy browser, click `block_xfer_inst` (`block_xfer`) to view the partially instrumented `wb_adr_o` bus. The values of the instrumented segments of the bus (bits 7 through 5 and bits 3 and 2) are displayed, while the uninstrumented bits are displayed as periods to represent an unknown value.

```

34 port (
35     Pwb_adr_o..010.10..: out std_logic_vector (addr_width-1 do
36     wb_dat_i: in std_logic_vector (data_width-1 downto 0) := (
37     wb_dat_o: out std_logic_vector (data_width-1 downto 0) := (

```

- Change the trigger position from middle to early using the early, trigger-position button on the left.



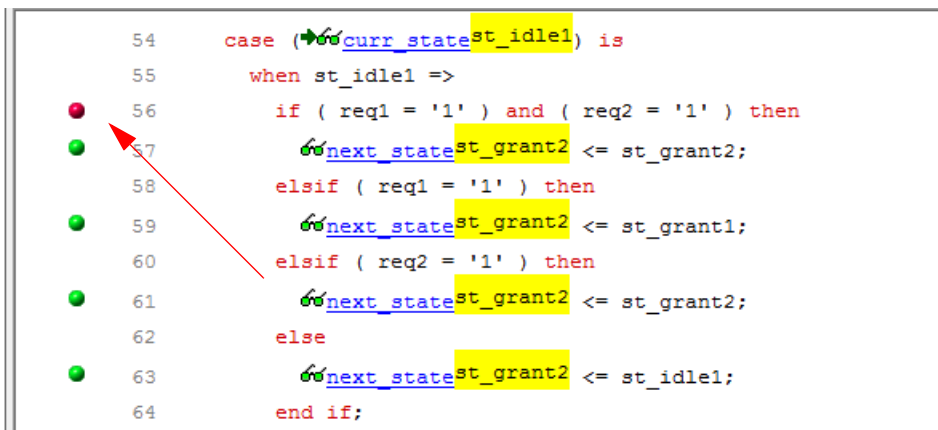
- Select `Debug->Run` from the menu or click the Arm selected IICE(s) for triggering icon (the running man icon in the menu bar) to generate a new trigger.
- Open the waveform viewer. The sample time remains unchanged, but the position of the entire waveform shifts so that ninety percent of the samples are available prior to the trigger (the red line) and ten percent are visible following the trigger.

Triggering on a Breakpoint

The next sequence switches from watchpoint triggering to simple breakpoint triggering. To setup a breakpoint in the arbiter:

- Close the waveform viewer if you have not already done so.
- In the hierarchy browser, click `arb_inst` (`arbiter`) to redisplay the arbiter in the instrumentation window.
- Click the `curr_state` signal and select `Watch` from the popup menu to toggle the watchpoint off. The lens changes from red to green indicating that you have deactivated the watchpoint trigger.

4. Click on the green breakpoint button below the case statement on line 56. The icon turns red to indicate that the breakpoint has been activated. This breakpoint is set to trigger when the curr_state signal has the value st_idle1 on the positive edge of the sample clock. Other breakpoint buttons can also be used for the breakpoint.



```

54     case (curr_state st_idle1) is
55         when st_idle1 =>
56             if ( req1 = '1' ) and ( req2 = '1' ) then
57                 next_state st_grant2 <= st_grant2;
58             elsif ( req1 = '1' ) then
59                 next_state st_grant2 <= st_grant1;
60             elsif ( req2 = '1' ) then
61                 next_state st_grant2 <= st_grant2;
62             else
63                 next_state st_grant2 <= st_idle1;
64             end if;

```

5. Now that you have an active trigger condition, arm the IICE trigger circuits by doing one of the following:
 - Clicking the Arm selected IICE(s) for triggering (running man) icon in the menu bar
 - Selecting Debug->Run from the top menu
 - Redisplaying the project window and then clicking the large Run button

Note: Using the Run button in the project window allows you to individually select multiple IICE units (necessary when performing cross triggering); the menu selections only arm the active IICE.

The IICE now waits for the trigger to occur. When the trigger occurs, the sampled data is transferred back to the debugger where it is displayed in the source code display next to the sampled signals as shown in the following figure.


```

54     case (curr_state st_idle1) is
55         when st_idle1 =>
56             if ( req1 = '1' ) and ( req2 = '1' ) then
57                 next_state st_idle2 <= st_grant2;
58             elsif ( req1 = '1' ) then
59                 next_state st_idle2 <= st_grant1;
60             elsif ( req2 = '1' ) then
61                 next_state st_idle2 <= st_grant2;
62             else
63                 next_state st_idle2 <= st_idle1;
64             end if;

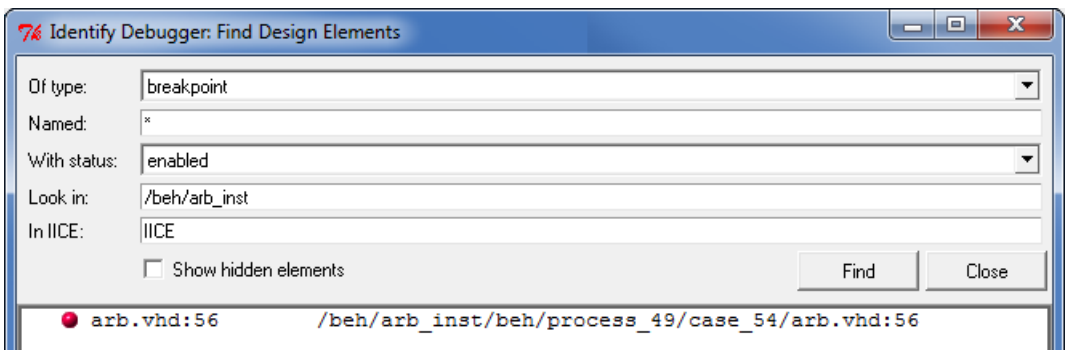
```

The values in yellow adjacent to the sampled signals are the data sampled from the FPGA (st_idle1).

You can check for active and inactive watchpoints and breakpoints by clicking the appropriate show enabled/disabled icon in the menu bar.

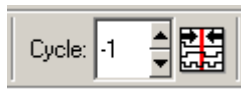


Clicking the show enabled breakpoints icon shows the single enabled breakpoint on line 90 for the corresponding process/case.



As with watchpoint triggering, you can change where the trigger point is in the buffer by selecting one of the Early, Middle, or Late icons and again arming the IICE. The trigger location changes the next time the IICE triggers.

The Cycle display in the middle of the menu bar shows the value zero. This is the point in the sample data buffer where the trigger occurred. By clicking on the up-down arrows on the right, you can increase or decrease the cycle count to display the data values before and after the trigger point.



Using the Complex Counter

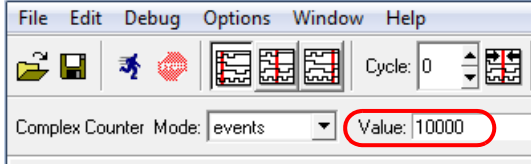
The tutorial design is instrumented using a 16-bit complex counter. Up to this point, the design has been debugged without using the counter. The counter is disabled by setting the counter mode to events (the default) and setting the counter value to 1.

There are four, complex counter operating modes:

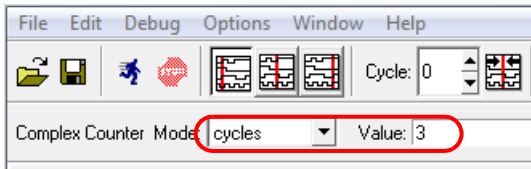
- cycles – triggers on value n number of clock cycles after the condition.
- events – triggers on the n th iteration of a trigger condition.
- pulsewidth – triggers when the condition has been active for n clock cycles.
- watchdog – triggers when the condition has been inactive for n clock cycles.

For this portion of the tutorial, the complex counter remains in the events mode and uses a large counter value to step through individual debug events. To use the complex counter:

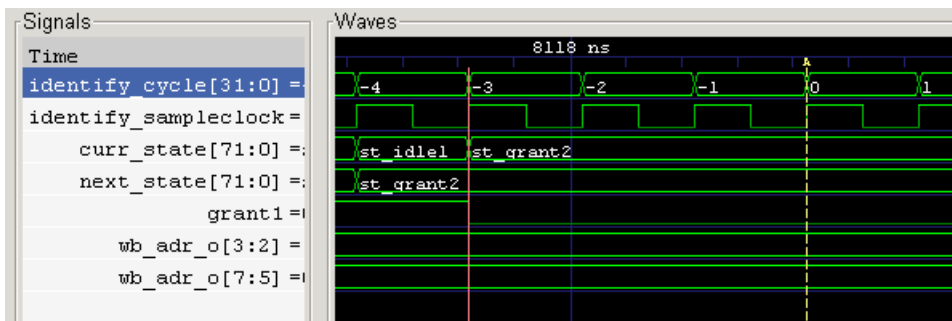
1. Deactivate the previously set breakpoint at line 90.
2. Click on the `curr_state` watchpoint and select Set trigger expressions from the popup menu.
3. Enter `st_grant2` in the left box (replacing `st_idle1`) and click OK.
4. Enter 10000 in the Value field to the right of the Complex Counter Mode field.



5. Select Debug->Run from the menu or click the Arm selected IICE(s) for triggering (running man) icon to arm the IICE. Note that this run takes a little longer to trigger as it must wait for the 10,000th event (st_grant2) to occur.
6. Set the Complex Counter Mode to cycles and enter 3 in the Value field as shown in the following figure.

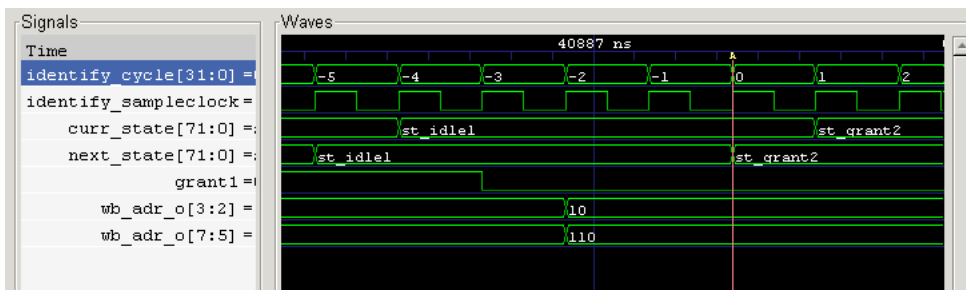


7. Click on the curr_state signal and select Set trigger expressions. Enter st_idle1 in the left field and st_grant2 in the right field of the Watchpoint Setup dialog box.
8. Select Debug->Run from the menu or click the Arm selected IICE(s) for triggering icon to arm the trigger.
9. View the resulting waveform and verify that the trigger occurs three cycles after the curr_state signal transitions from st_idle1 to st_grant2.



10. Close the viewer.

11. Set the Complex Counter Mode to pulsewidth.
12. Set the counter Value to 5.
13. Click `curr_state`, select Set trigger expressions, and enter `st_idle1` in the left field; leave the right field blank and click OK.
14. Select Debug->Run from the menu or click the Arm selected IICE(s) for triggering icon to arm the trigger.
15. View the resulting waveform and note that the trigger occurs on the fifth rising clock edge after the `curr_state` signal transitions to `st_idle1`.



16. Close the waveform viewer.

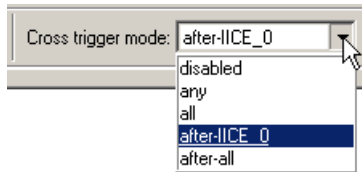
Cross Triggering

Cross triggering (triggering between two IICE units to debug logic across clock domains) is described in this section. The design used with the development of this tutorial has two clock oscillators, a 35 MHz oscillator and a 50 MHz oscillator. Signals between the arbiter and the other logic are synchronized in the design with these clocks.

For this section of the tutorial, you will use `IICE_0` which you previously instrumented in the `block_xfer` module and which is clocked by the 50 MHz oscillator as a trigger for the first IICE unit (IICE). This first IICE unit was instrumented in the arbiter module and is being clocked by the 35 MHz clock. When the arbiter sees the request from the `block_xfer` module, it changes to state `st_grant2` to process the request. Triggering the IICE module occurs when `IICE_0` triggers on the request and the state changes.

To demonstrate cross triggering between clock domains:

1. Select the IICE (left) tab and set the Cross trigger mode to trigger on after-IICE_0.

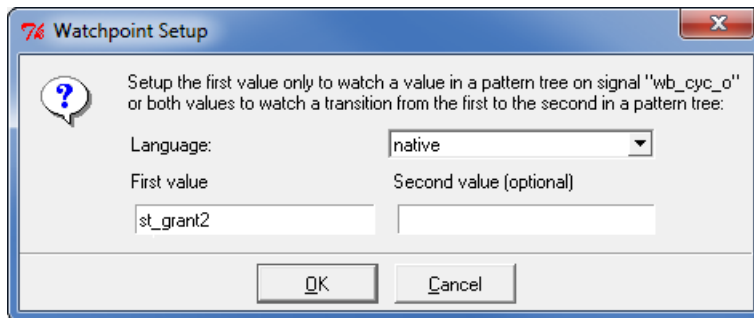


If the drop-down menu is blank, the Allow cross-triggering in IICE checkbox on the IICE Controller tab for IICE unit IICE has not been checked in the Identify instrumentor and you must go back to the Identify instrumentor and enable the checkbox. Click File->Save activations to save your current settings before exiting the Identify debugger

2. Set the Complex Counter Mode back to events and enter 1 in the Value field.

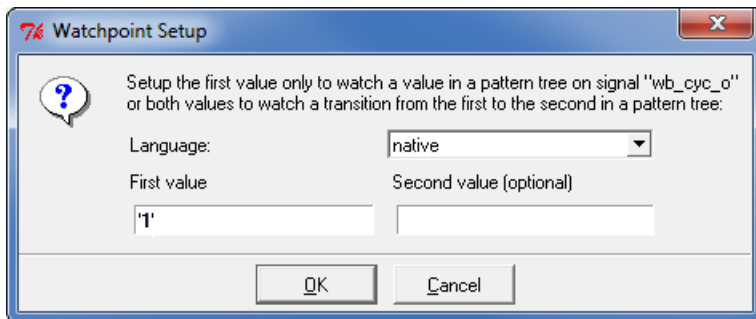


3. Click on the curr_state signal on line 45 and set the trigger expression to st_grant2, and click OK.



4. Click on the IICE_0 tab and verify that the Cross trigger mode is set to disabled.
5. In the hierarchy window, expand the beh block and select blk_xfer_inst (block_xfer).
6. In the instrumentation window for IICE_0, click on wb_cyc_o on line 38 and select Set trigger expressions from the popup menu to display the

Watchpoint Setup dialog box. Enter '1' (with the single quote marks) in the first field and click OK; the watchpoint icon turns red to indicate that it is enabled.



- Open the project window by clicking the bus_demo tab. Select the Run button to arm the two IICE modules. IICE_0 triggers itself when the request is active and also triggers the first IICE unit (IICE). When this IICE triggers, the run is complete.

Note: Using the Run button in the project window allows you to individually select multiple IICE units (necessary when performing cross triggering); the menu selections only arm the active IICE.

- Click the IICE tab and note that the run has completed and that values are annotated in the code.

```

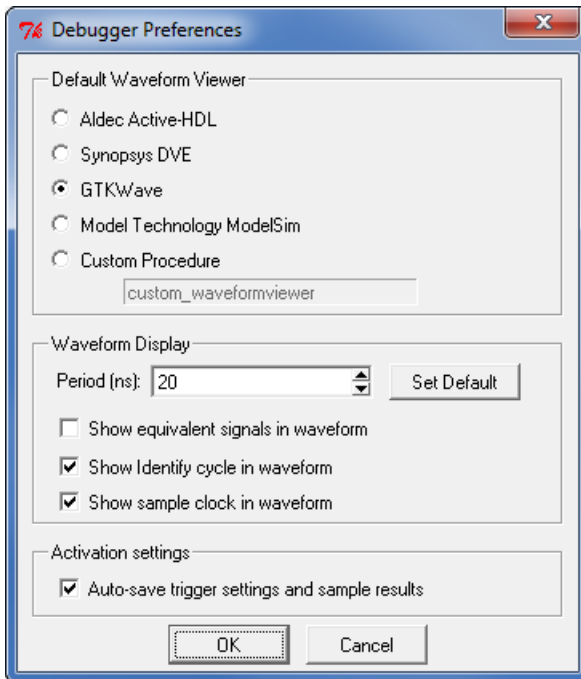
38   grant1'1' : out std_logic;
39   grant2 : out std_logic
40   );
41 end arbiter;
42
43 architecture beh of arbiter is
44   type states is ( st_idle1, st_idle2, st_grant1, st_grant2);
45   signal curr_state st_grant2, next_state st_idle2 : states :=

```

- Advance and retard the cycle count using the up and down buttons to see the values of the instrumented signals change.
- Close the debugger. The lab is complete.

Generating Waveforms

The data captured from the design can be displayed as waveform data by using a waveform viewer. Viewer setup is controlled by the Debugger Preferences dialog box. Selecting Options->Debugger preferences from the menu bar brings up the dialog box shown below.



The Period field sets the period for the waveform display and is independent of the design speed.

After running the Identify debugger, the waveform viewer is displayed by selecting Window->Waveform from the menu or by clicking the Open Waveform Display icon in the menu bar.

All sampled signals in the design are included in the waveform display. Two additional signals are automatically added to the top of the display. The first signal, `identify_cycle`, reflects the trigger location in the sample buffer. The second signal, `identify_sampleclock`, is a reference that shows every clock edge.

